Computational Fluid Dynamics

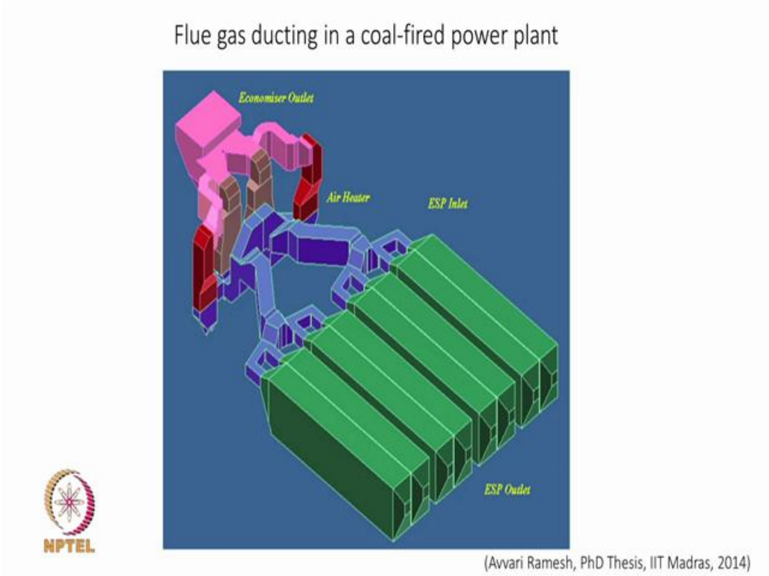
Prof. Sreenivas Jayanti
Department of Computer Science and Engineering
Indian Institute of Technology, Madras

Lecture – 01
Lesson 1- Motivation

Welcome to the course on Computational Fluid Dynamics. It is a course which is intended for students who are in engineering disciplines like chemical, mechanical, aerospace, civil engineering in which fluid mechanics is very important ah phenomenon in the equipment and processor that they deal with. It can also be of interest to scientist, practicing engineers who need knowledge of fluid mechanics in order to design the process better, in order to understand the phenomenon better.

Now, when we come to fluid mechanics, it is very old subject. We have known it for hundreds of years, but knowledge of fluid mechanics is limited by the inherent complexity of the phenomenon and of the equations that govern fluid motion. The equations which describe fluid motion as we use them today have been known from more than 100 years, 150 years in fact, but there is no analytical solution that is known to this equation for the general case and what we study in our fluid mechanics courses are only for the simplest possible situations like fully dial flow in a straight pipe and may be some very simple one dimensional flow cases like that, but when we look at practical applications that engineers and scientist deal with, these kind of exact solutions are totally inadequate and when you look at how we can design them, there are number of design correlations, but the design correlations also have lot of limitation about these things. We will take an example of practical fluid mechanics problem and see how inadequate state of knowledge is in order to tackle this practical problem in a proper way. So, we will look at the specific case of a fluid gas ducting in a coal power plant, coal fired thermal power plant.

(Refer Slide Time: 02:31)



This is a typically example of how fluid flow occurs in many process industry applications. What we have here is part of the gas ducting. Above this is a large duct which carries hot air hot fluid gas from the furnace and it enters here. At the time it enters here, it is still at some 500 degree centigrade. So, there is a lot of thermal content within this and this needs to be exploited, so that the overall efficiency of the thermal power plant is high.

So, what is done is that this hot gas is sent into two heat exchanges. This one in red and this in brown are heat exchanges and these are air free heaters. So, the hot fluid gas will heat up the incoming air which is used by combustion and there by exchanges some of the thermal energy which would otherwise be lost. So, in order to make this happen, the fluid which is coming in through this rectangular that of a cross section of a rectangle like this, it is a vertical pipe of this cross-section is split is may try into 90 degree thing and then, split into two strings. One going left and one going right and this stream here is again split into two streams; one going through a primary air heater and one going to through a secondary air heater.

So, this goes in here. It gets a cold in the process of heating of the primary thing and then, this stream goes down here and this stream is going to the secondary air heater. So, this goes in here. It gets cold in the process of heating of the primary thing and then, this stream goes down here and this stream is going to the secondary air heater and then, it comes through like this. They both mixed and then, coming to this and this section will lead them into the electrostatic precipitator e s p and the e s p has a number of plates in which the dust which is there in the fluid gas is collected by electrostatic phenomena.

So, in order to the difficulty here is that we have a duct here and this has to be spread to this battery of four electrostatic precipitators. These are very huge compared to this. So, means that will have to create space to accommodate these electrostatic precipitators. The reason why these are huge is that here the velocity can be high of the order of 10 to 15 meters per second, but from best operation inside the electrostatic precipitator, we need to have something like one meter per second velocity which means that the velocity has to be increased and the same fluid it has to maintain because it is a continuous flow system and may at the cross-sectional area of the e s p should be very high. So, this should be very large in order to any be ten times as large as these in order to have the same reduce velocity of one meter per second.

So, now comes the engineering difficulty of not distribute the flow which is coming from list into this very large equipment. So, you have to increase the size. So, you also need to accommodate four of them. So, that means that you take a bend here and then, you come back to an angle and then, you split into four streams here and this stream goes into this stream goes into this and here it goes to this and when we are looking at thermal power plant, it has to be operational continuously, but we also need to look at the maintenance issue.

So, that is why we need to have duplication. So, the two are duplicated here, so that we have four of this and then, you have an arrangement here with the gate wall. If all the four are working, then two of them will be taking the load here and two of them will be taking the air here, but if one of them is in under maintenance, then you open the gate wall here and then close it down here, so that all the air coming from here and here is spread through this and then, these two are working.

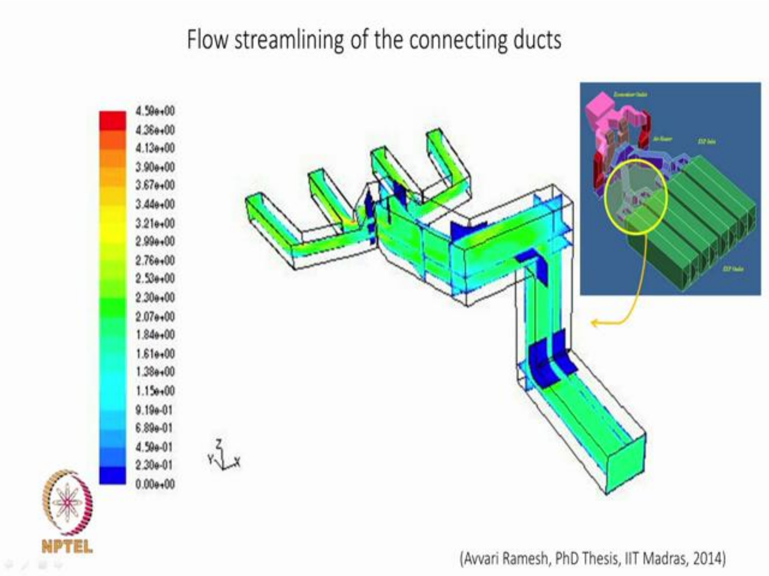
So, we can have a situation where only two of them are working full time to remove all the dust from the fluid gas or all the four of them are working. So, need to have a provision for it. So, you need to have all those provisions which will make the design very complicated in the sense that if we now try to design it and then, if you want to apply your know correlations and analytical solutions for this, you will see that its practically impossible because if you now look at it from design point of view, we have rectangular duct which is bringing in hot fluid gas into this and it has to take an immediate 90 degree bend and this bend is not like the smooth kind of large radius bends that we consider. It is not an albo or the standard type. It is of a specific type. Why is it of specific type? It is because the dimensions here are very large this this duct here can be something like 4 meter wide and 6 meters tall.

So, that is a huge duct. So, this can be like this whole length can be something like 25 to 40 meters. So, looking at such huge equipment and these are made in terms of as rectangular walls which are erected on site and that means that since is a huge duct, you do not want to carry them very long because that link is costlier and it will also increase the running cost and it will also increase the land cost because unit accommodate all these things in land, in public land and land will have value. So, in practical problems you cannot honor the kind of l by d areas that are usually required for a specific correlation to work or for some analysis assumptions like fluid flow to become applicable.

So, you need to deal with the practical compulsions. So, you would say it takes a rapid close bend here and from here it has to split into two. How do you split into two? Analytically very difficult, it is very difficult to predict what would force the fluid which is coming from this and going through a bend sharp, bend like this to split even ally between these two and then, as its coming through this at some angle, again it has to split through this and the fluoride through this and fluoride through this are not necessarily the same. It depends on the process requirement how much heat you want to give to the secondary air free heater and how much you want to give to this primary air heater. So, that means that it cannot be tackled in a simplistic way. We have to take, you need to accommodate these air free heaters and we have ducting which is bringing fluid into this and separate into this and then, those are happening, those ducting details are not shown. So, that means that we have complicated irregular kind of piping layout in which you have fluid coming in taking sudden turns and then splitting and then further splitting equally between these two and splitting unequally betting between these two. Going through a rotary heat exchanger region rate, heat exchanger here the geometry of which is very complicated and then, it comes back here and then, this comes back and in mix together and then, they go through again come up and then take bend and then go through this angle thing, split up here.

So, you can have a lot of complexity which is associated with the case of single phase flow in a practical situation like the power plant and there are design because if you look at the electrostatic precipitator, there are four of them here and with two in nets, each you would like all the four to be equally fed under normal operating conditions. So, that means that the fluid through this must be the same as the fluid through this and the fluid through this must be the same as the flow through these things. They must be equal split here and you would also like to have despite end because of the sharp turns and then, fluid splits and all that you would like to minimize the pressure drop as much as possible. So, it is like to control the flow, regulate flow despite all these singularities and abstractions that are coming here.

(Refer Slide Time: 11:49)

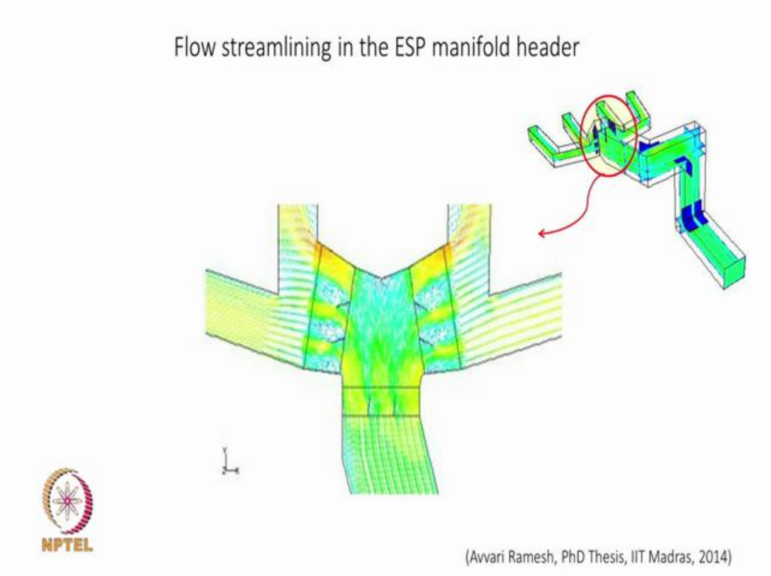


.

So, if you look at what can be done is we are looking at just this ducting here which is taking the combined flow after it goes to the true air free heaters and then, it is spread into this e s ps. So, we are looking at how to make the fluid distribution even at the four exceed points here, at the four exceed point despite the facts that is coming through bends here, bend here and bend here angled in and all these things.

So, one of things that is often done in industry in order to reduce the pressure drop and create flows steam lining is to put some guide plates. So, here we have guide plates, here in guide plates, here in this, then there is a guide plate and here you have guide plates and the idea is that you want to put this guide plates properly in order to get a good distribution. So, this becomes a totally unknown kind of a situation for which we are not equipped from knowledge point of view. As engineers, we do not know where to put these things we can use. So, what engineers have developed over the years and decades is to have some thumb rules for locating these things, but how do you know that these thumb rules will apply for this particular situation with these kinds of layouts and where do you put them and how do you make this uniform.

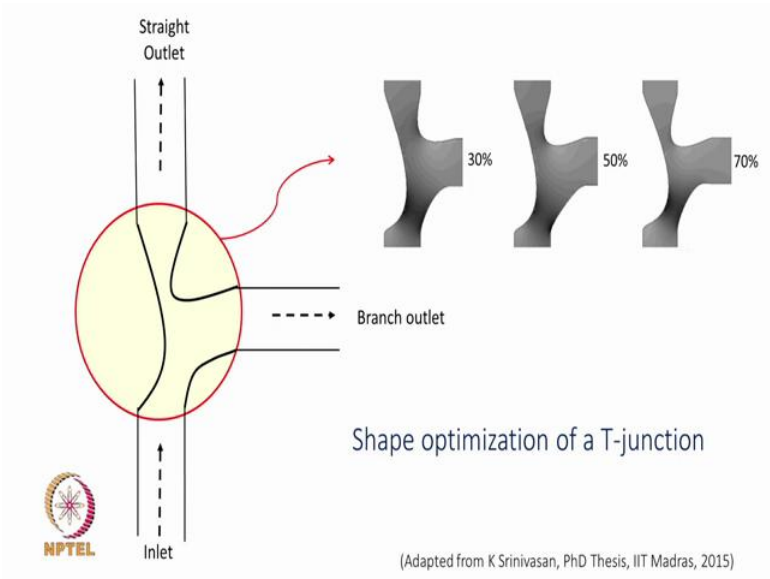
(Refer Slide Time: 13:22)



So, if you look at the next thing here, you are concentrating on just this junction here, the manifold header into the e s p and the flow is coming in like this. It should split evenly into these two and then, the same amount of flow should be going through this in this, so that each e s p electrostatic precipitator is fed the same amount of dusty gas. So, how do you persuade this one which is coming in at some angle to split between, evenly between the left side and right side and after it turns here, how do persuade it go through this and this equally because this length is shorter and this length is longer and as its coming through taking a bend, it already has some previous history of some inclination to go either this way or this way.

So, you can put guide plates here, you can put some guide plates to correct the imbalance between this flow and this flow and then, you can put some guide plates to persuade the flow to go into this. On what basis do we put the guide plates? To do this experimentally would mean that we have to build this whole thing in our lab and then, try various things and they will take months and it will be very expensive. So, this is what we are actually looking from fluid mechanics. We are looking from fluid mechanics the possibility of controlling the flow in practical situations, so that we can derive the intended benefit from the process. The process can be done optimally efficiently at another example that we can consider.

(Refer Slide Time: 15:03)



Let us say that we have a simple t conjunction flow is coming in straight to this and it has to split into side branch and it has to go into the straight branch. Usually the process requirement will tell us how much you want to go into the straight and how much you want to go into the branch, but what can be done as design engineers, we design a junction here in such a way that the process, the flow which is coming in will go automatically into this and automatically into this. We cannot do that because we do not know what should be the shape we can draw curves like this. They look very interesting curves, but how do we know that this would actually do a job using computation fluid dynamics. It is possible to solve the exact equations which govern the flow through this t junction.

That means that it is possible to say that if this is a geometry, this is the width of the direct turn length, this is an orientation this side branch and this is width or a slide branch and all things is possible using CFD to solve the equations which describe the flow through this inlet going through this outlet and going through this out let in such a way that we can come up with a junction shape which will automatically deliver which will direct the flow require amount of a flow into the branch outlet without having to have any walls.

For example, here we have three different shapes. This particular junction shape here will deliver 30 percent of the flow into the side branch and 70 percent will go through the straight branch and this one will go will have 50 percent going through the side branch and this will go 50 percent here and this will actually give a 70 percent going through side branch and this will be going only 30 percent and what are we doing here is, we are creating a constriction here, we are necking this region by changing the shape of these things in such a way that more of this is going through this and less of this is going through this. In fact, if we do not have any of this control, if we just have straight sections here, 90 percent of the flow would go through this straight section and only 10 percent would come through this.

So, if you want to have more flow going through the outlet, you constraint, you constrict it, so that you direct it to go into this. Exactly how much we have to direct is something that is not known. We can make a sketch as engineers, but how do we know that it is going to be quantitatively correct using CFD using by solving the exact equations correctly. We can make this kind of designs prospect. So, that is the power of CFD that is what we are looking from as a design engineer. This is what we like to be doing, but this is not really possible with our simplistic flow analysis tools that we enable in our regular engineering programs. So, if you have done only one or two courses in fluid mechanics, you would not be able to tackle these realistic situations, but computational fluid dynamics in which we are not looking at approximate solutions using correlations in all that, but we are trying to solve the exact equations which govern the fluid flow, enable us to do an exact treatment or a fluid flow problem, so that we can generate, we can hope to tackle these kind of industrial situations where you have t junctions and you have this kind of flow, difficult flow situations which cannot be tackled with using conventional flow knowledge that we have.

So, that is the motivation for us to do c f d. So, to retreat why we want to do, why we want to do this particular course and why we want to learn about computation fluid dynamics is because practical flow situations are much more complicated geometrically and physically than what we recruit to do in our based on our knowledge given in the courses theoretical courses that we go through.

So, although there are thumb rules for designing in practice for any of these things, we cannot hope too much. For example, we using thumb rules, we will not be able to tackle this t junction problem and then, come up with shapes of this junction which will give you 30 percent or 50 percent or 70 percent. If you want to do that, we have to do either experimentation which is very closely and which is very time taking or we have to try to solve the governing equations, but the governing equations are very complicated in the sense you do not have a single equation. We have at least a minimum of four equations and each equation is a partial differential equation, three of them are second order equation and one of one of them is first order equation in the simplest case and the equation are also of mixed character like hyperbolic, parabolic, elliptic and these kind of things which make the solution very difficult.

So, it is not the lack of knowledge of the equations, but it is the difficulty of getting an analytical solution in a practical cases that makes it extremely difficult for an engineer or a scientist to tackle real world problems, but if we can do the equations, solve the equations numerically and if we can solve the equations in fully developed three-dimensional flow geometry, then it will be equipped with the tool that we can use for a design and CFD offers that kind of tools because CFD a knowledge of computational fluid dynamics techniques will enable you to go straight to the exact governing equations and then, solve them in the geometry in the irregular geometries like what we have seen and with the kind of boundary conditions and with the kind of interventions that you want to do.

So, it would be possible to simulate the flow through realistic flow equipment by doing, by going the computation fluid dynamics way and that provides us the motivation for this course. Computational fluid dynamics is not an easy subject. It has inputs, it derives from knowledge in different disciplines obviously in fluid mechanics, but there is also lot of mathematics that goes with it and there is also lot of programming and computer science, mathematics, algorithms and data structure. Although things will be going will be needed in order to make very important to something which can be used by lots of people. We are not going to discuss all those things. In this course we are going to discuss the basic concept involved in the solution, numerical solution of the governing equations. So, the outline, the course plan is what is given here.

(Refer Slide Time: 23:15)

We are looking at a twelve week course. Within this I would like to give you, I would like you take you through CFD approach to this solution governing equations and I would like to touch upon all the important concepts which enable us to get to the solution of this. So, we have twelve weeks including this particular week. In the first two weeks, we are going to look at CFD and it is simplest form. So, in the first week we are going to look at how to calculate flow in rectangular that if it is circular pipe we already know under laminar flow conditions, it will have parabolic profile, but it is a rectangular duct. It is not so easy.

So, we will see how we can do it, how we can get the flow field in fully developed flow through rectangular duct using the techniques that we are going to develop in further weeks of a computation fluid dynamic. In the second week, we are not going to look at rectangular duct, but a triangular duct which possess additional changes because rectangular duct can be described in a circular can be described in cylindrical coordinates, but how do you describe a triangular. So, we bring in another fair set of computational fluid dynamics to deal with this. So, by the end of two weeks which is the first model, we have the basic idea of what CFD is, but we do not have enough knowledge to be confidence about it. The concepts are still hidden.

So, in weeks three and four, we are going to derive the equations, the exact equations which govern the fluid flow which we need to be solving. This is not usually done in the first course of fluid mechanics. So, I am going to derive them, so that we have good understanding of what the equations are and how they come about and what is the basis for CFD and after that we are going to this third model in which we are going to bring in the basic concepts of CFD which will help us solve this partial differential equations. So, that is in week one, we are going to deal with the finite difference approximations and how we can use those things to convert partial differential equations in to algebraic equations and in the week after that, we are going to look at some other properties of this discretization and concepts like consistency, stability and convergence which actually help us in determining what is the best way of going about this finite difference approximation to get to the solution and in the next two weeks, in the fourth module we will be looking at not just the solution of one equation, but we look we looking at the set of coupled equations which will be needed to deal with three-dimensional flows.

So, we are going to look at the compressible flow case and then, the incompressible flow case. So, the incompressible flow would be much more of interest to civil engineers and some mechanical engineers, some chemical engineers like that and incompressible flow should be more for process industries, but both can be found in all kinds of applications and the first module is more about mathematical techniques which are required for the solution of the linear equations, algebraic equations which are the equations that we are really solving in computer and will be solving tens in tens of thousands or millions of these equations.

So, we need to be very good at solving this. So, the kind of specialist techniques that have been developed in the CFD with relation to reduce our solution of linear algebraic equations are what we going to do in the next two weeks and in the last two weeks which is the final module. We are going to go to the finite volume method and formulate a problem and we discuss how we can generate grid for this and then, how we can solve these things using the finite volume method, not using the finite difference method which is what we are going to introduce in week five.

The last week we will spend on an introduction turbulent flow, turbulence modeling which is required to tackle real world problems when the flow is turbulent which often case in many process industries is. So, we would like to be touching a bit about turbulent flows and turbulence modeling which is required for practical cases. So, this is the essence of the course plan. Each week will have a assignment and there will be discussion, there can be a discussion of the assignments, other concepts in the forum, CFD online forum and the assignments will be have a deadline. You will have to submit at the end of that and then, there will be an examination which is conducted towards the end of the course. More of those details will be discussed in the next lessons.

So, what we have learnt in this lesson, in the first lesson is why we need to do study CFD and why we need to go into that because practical flow situations as shown in this figure are very different from the kind of the assumptions that we need to make in order to get an analytical solution or apply thump rules and practical solutions require a lot of optimization getting it correct, so that we can operate efficiently and these kind of things require the possible degree of solving the exact equations correctly and that is what is needed. That is what is made possible using the computational fluid dynamics approach and that is why we are studying this.

Thank you.

We will meet again shortly for the second lesson.